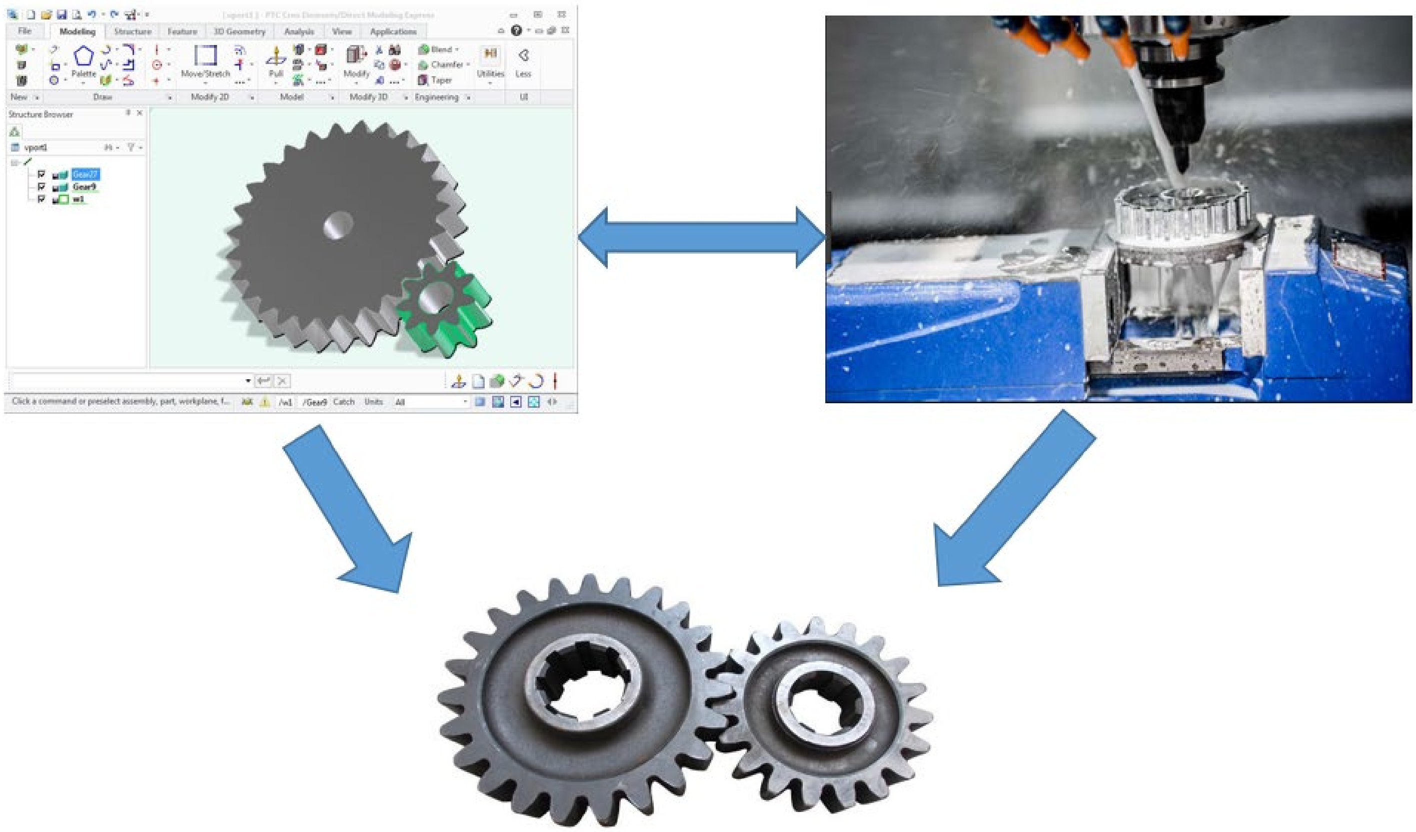
LA-UR-18-31872
Approved for public release; distribution is unlimited.

Title:	Design Considerations Which Can Simplify Manufacturing (U)
Author(s):	Vest, Erwin Glen
Intended for:	Reference document to be used to guide new engineers at LANL.
Issued:	2018-12-21



Design Considerations Which Can Simplify Manufacturing (U)

Earl Vest

PF-WFS R&D Engineer

10/25/2018

LA-UR-18-xxxxxx

This document has been DC Reviewed and was deemed to be **UNCLASSIFIED**

DC Reviewer: Danny A Martinez, PF Division DDL, 10/25/2018

Derived From: CG-LANL-COMP-2, 3/17, DOE-OC

Abstract:

The purpose of this paper is to recommend design considerations which can significantly reduce the time required to manufacture parts, reducing cost to manufacture parts while increasing the quality of manufactured parts and minimize the likelihood of nonconformance conditions. The product design is ultimately the responsibility of the project engineers and their design teams. This paper is not intended to supersede design requirements.

Background:

The production of any product begins with the design of the product. How the product is designed will dictate how it will be made. Consequently, the design of a product (or part) has a major influence on the ultimate cost to produce and deliver the product to the customer. The graphic in Figure 1 below illustrates the influence that the design organization has upon the cost, quality, and cycle time for a product versus the influence the manufacturing organization has on these factors.

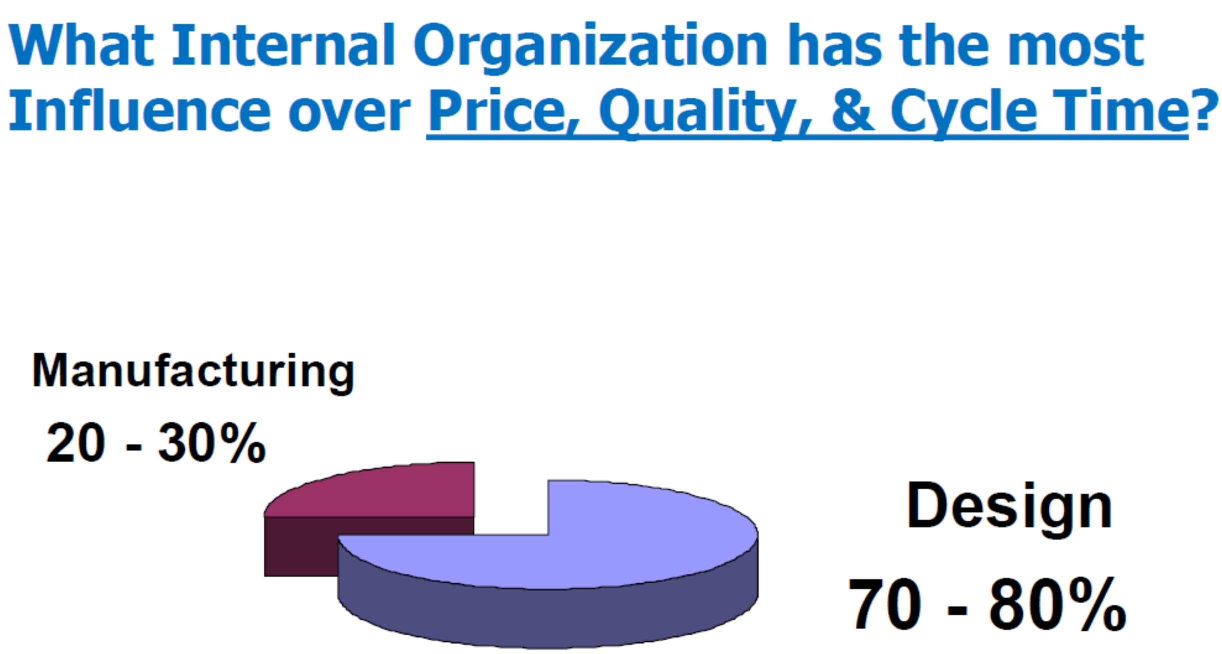


Figure 1 – Source: gatech.edu/files/capstone/L071ME4182DFA

PF Division is responsible for manufacturing the majority of components that make up the assemblies for hydro tests, sub-critical experiments, etc. Because of the large number of parts PF Division has to fabricate during the FY to meet program schedules, it is important that PF Division produce parts in a timely manner, while trying to keep costs low and quality high. There are a number of factors that impact PF Division's ability to meet or exceed customer expectations. This paper discusses ways in which design organizations can help the manufacturing organizations (PF Division, etc.) produce components that are high quality at the lowest cost possible.

Design Considerations which can make manufacturing easier:

1. **Choosing Material:**

- When choosing a material from which to machine components, allow the use of different forms of the material such as bar stock or plate. There can be significant differences in the cost and lead time for acquiring different forms. The table shown in Figure 2 shows the approximate cost per lb of common metals, their machinability ratings, and tensile strengths.

Alloy, Temper and Spec	Machinability Rating (1212 steel is 100%)	Ultimate Tensile / Yield Strength (ksi) typical	Price in $/lb. (Feb 2009)
6061-T651 Extruded Bar - ASTM B221, AMS 4150, QQ-A-200/8	320%	45 / 40	$1.60
6061-T651 Wrought Plate – ASTM B209, AMS 4027, QQ-A-250/11	320%	45 / 40	$3.20
2024-T351 Extruded Bar – ASTM B221, QQ-A-200/3	380%	68 / 47	$3.15
2024-T351 Wrought Plate – ASTM B209, AMS 4035, QQ-A-250/4	380%	68 / 47	$3.95
7075-T651 Extruded Bar – AMS 4154, QQ-A-200/11	340%	83 / 73	$3.35
7075-T651 Wrought Plate – QQ-A-250/12	340%	83 / 73	$4.25
MIC-6 Cast Aluminum Plate – (Very Stable)	340%	24 / 20	$3.25
304 Stainless Bar – ASTM A276, ASTM A479, AMS 5639, QQ-S-763	45%	90 / 40	$1.45
303 UNS Stainless Bar – ASTM A314, ASTM A320, ASTM A582, AMS 5640	78%	90 / 35	$1.89
416 Stainless Bar – ASTM A314, ASTMA582, AMS 5610	110%	75 / 40	$1.65
17-4 PH Stainless Bar – ASTM A564 Type 630, AMS 5643	48%	150 / 110	$2.25
1018 Steel CF Bar – ASTM A108	78%	67 / 45	$1.10
A36 Steel HR Plate – ASTM A36	72%	(58-80) / 36	$1.10
12L14 Steel Free Machining Steel Bar – ASTM A108	193%	78 / 70	$0.80
4340 Alloy Steel Bar (annealed)– ASTM A322, ASTM A304	57%	110 / 66	$1.40

Figure 2 – Source: Pro CNC Inc.

- Consider the strength versus machinability rating as well when choosing a material. For example, some aluminum alloys can have better performance than some grades of steel and are easier to machine (higher machinability rating).
- The per-part price for raw stock can be reduced by buying the stock in bulk quantities. For example, if several components for a project can be manufactured from the same raw stock, purchasing a larger quantity of the material can be more cost effective than purchasing smaller quantities. Also, some raw stock is sold by common section sizes, for example, steel pipe may be sold in standard 10 or 20 foot sections. If a 2 foot section is needed to make a part, the distributor will cut what you order from a standard section and then charge you for the entire section. They will then keep the “drop”.

2. **Geometry Considerations:**

- **Radii –** When designing parts that have pockets or other features with vertical inside corners machined into them, you will need to leave a radius in the corner because the machining process uses rotating tools to machine the feature. Use the largest radii you can. Figures 3 and 4 provide examples of two types of radii and the type of tool used to cut them.

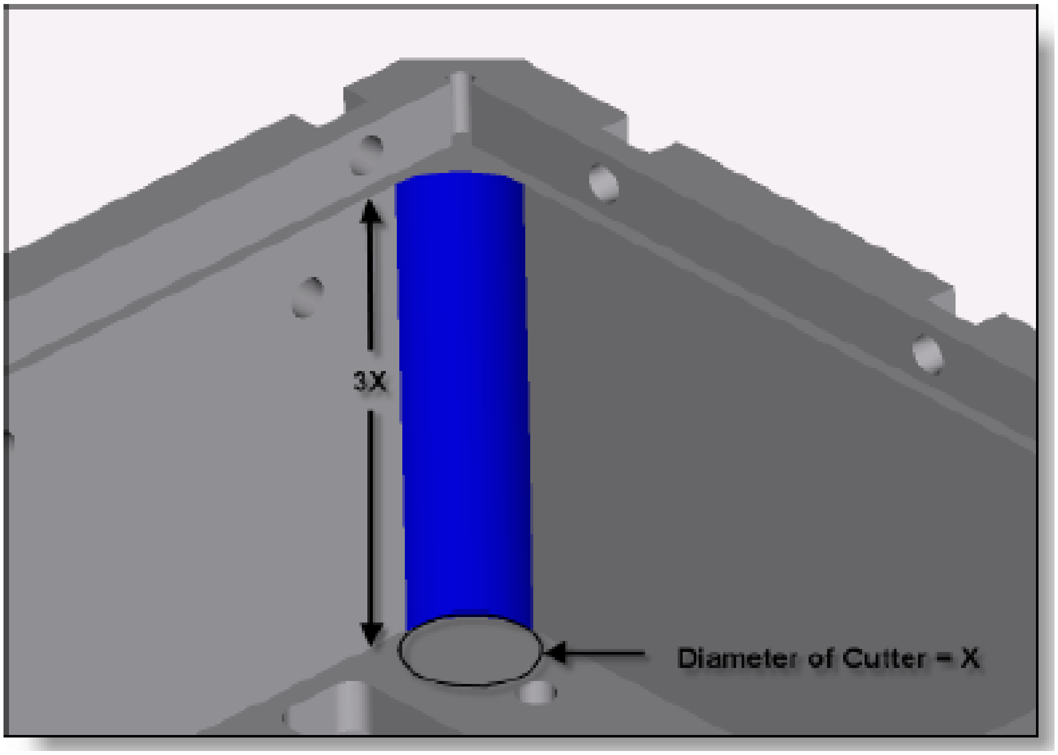


Figure 3: Short and Rigid tool can cut this. Source: Pro CNC Inc.

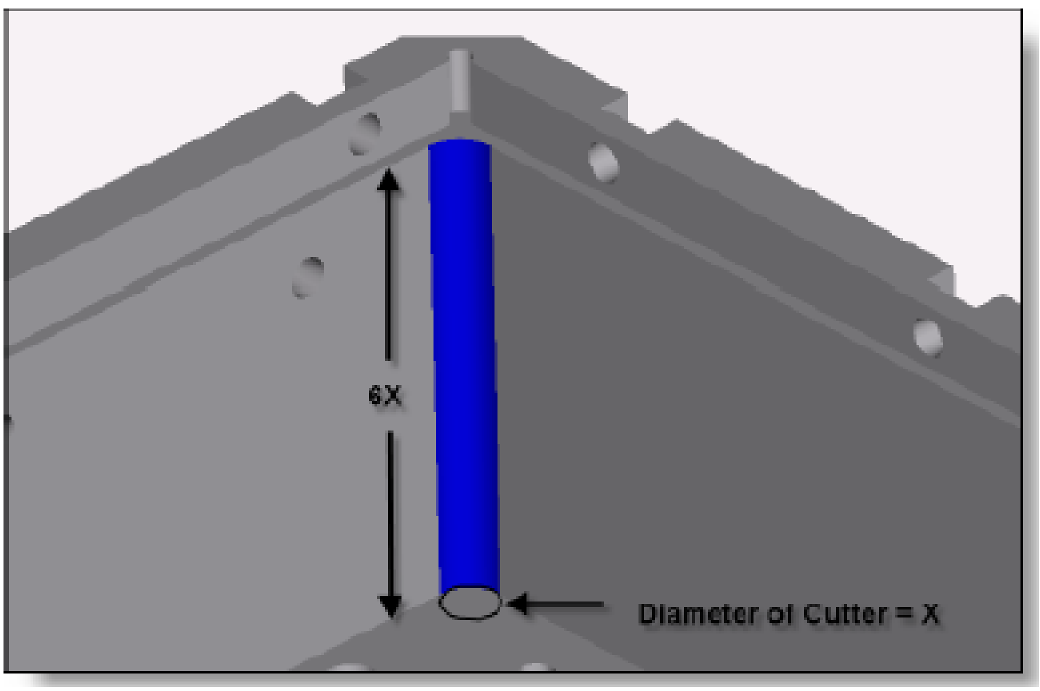


Figure 4: Long and Flexible Tool Required. Source: Pro CNC Inc.

The tool used to machine a particular radius will have a diameter of 2 times the radius you put in your model. If the part is designed with a 1/8” corner radius, a maximum 1/4” diameter tool may be used to machine the radius. The larger the tool that can be used in a corner, the faster it can feed through the material. As the length of that corner increases, the length of the tool must also increase and must be fed much more slowly to avoid deflection and breakage. For each doubling in length, the feed rate is cut by more than one-half. Therefore, assume that double the ratio equates to double the cost of that feature. A good ratio is less than 3:1. Ratios of 4, 5, or 6 to 1, result in much slower feed rates. The uppermost limit is 8:1, which is very slow and expensive to machine.

- If a long small radius is required, consider utilizing a virtual sharp or small corner radius. Figure 5 illustrates how a virtually square corner can be made with very little intrusion into the wall.

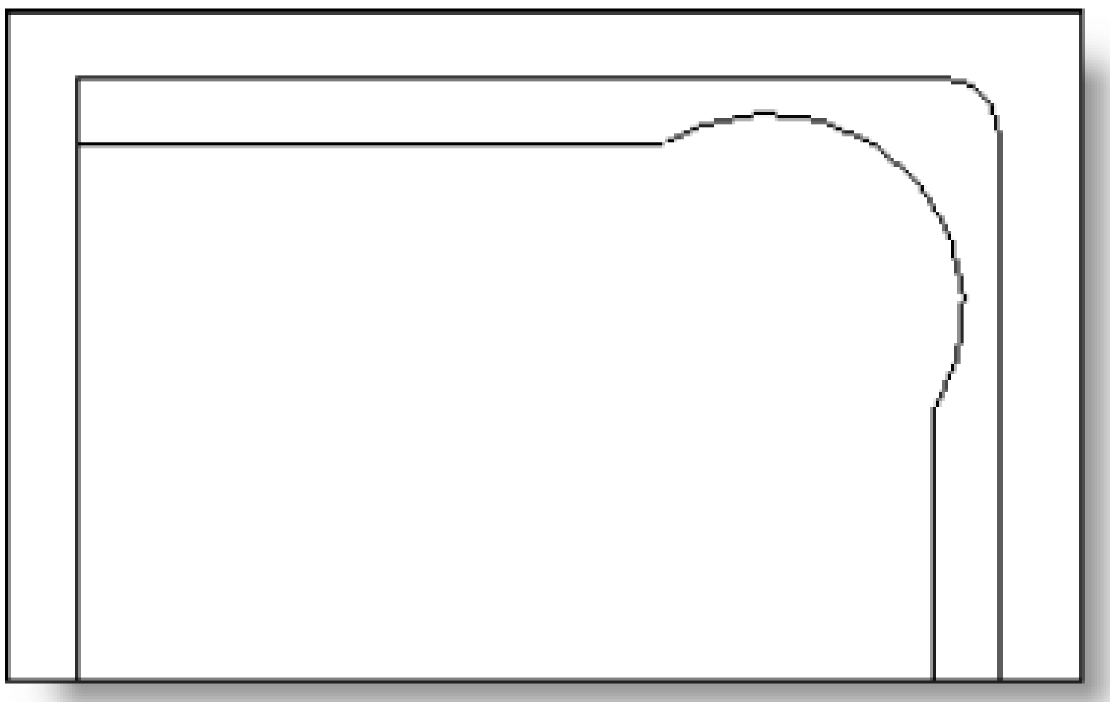


Figure 5: Virtual Sharp or Small Corner Radius, Source: Pro CNC Inc.

The key is to not put the center of the radius on the intersection of the inside edges. Put the center inboard of the edges and adjust to fit specific applications.

- Model floor radii smaller than wall radii. See Figure 6 for example. When a design calls out equal floor and wall radii, two tools are necessary to clean the area completely. The wall will have to be cut with a ball end mill while the floor has to be cut with a flat end mill. This will leave a triangular shaped section that only the ball end mill can remove. See Figure 7 for illustration.

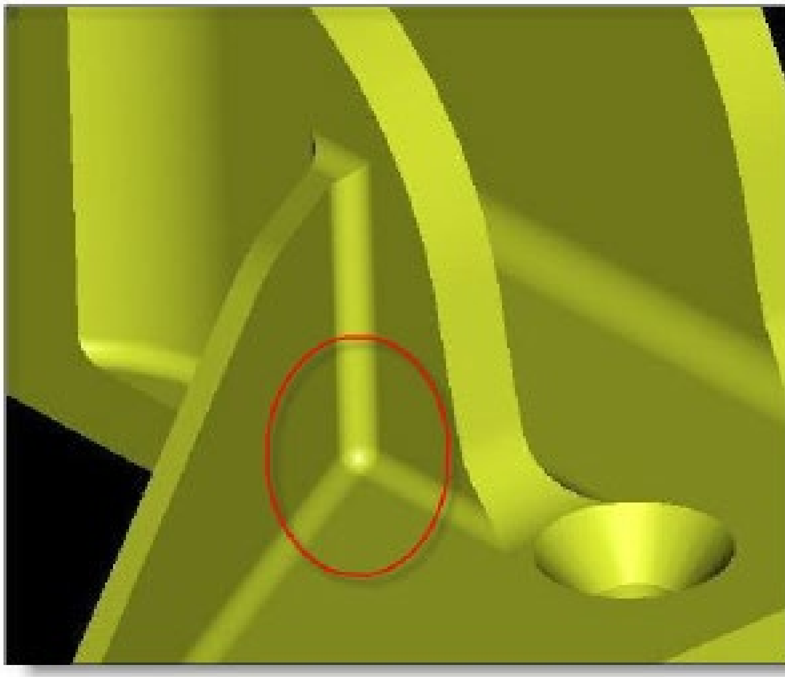


Figure 6: Equal Radii on floor & wall cost 10X more to machine, Source: Pro CNC Inc.

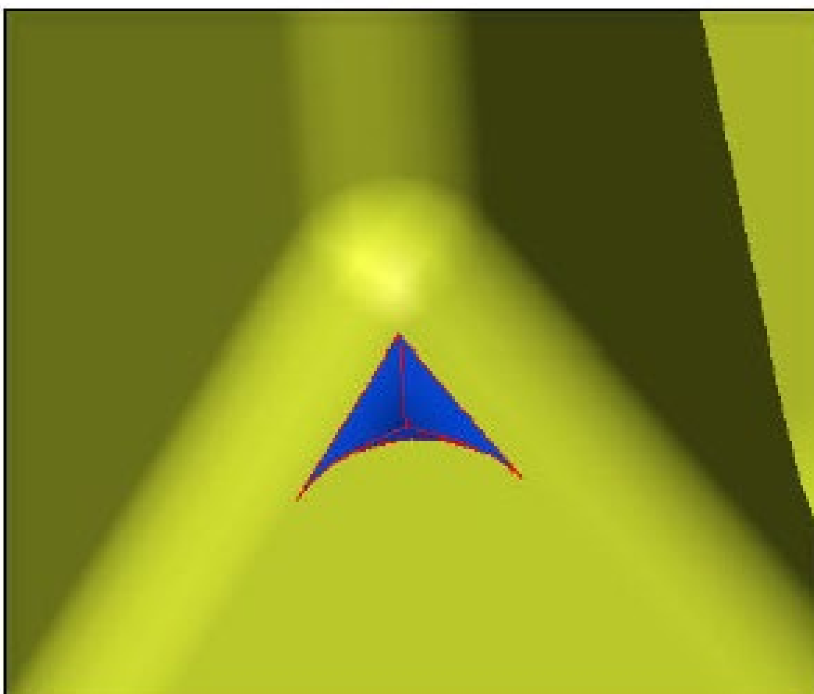


Figure 7: Area in Blue will be hard to remove. Source: Pro CNC Inc.

This can be avoided by making the floor radii smaller than the wall radii. The machinist can machine the entire area with one tool that has a flat bottom with radii on its tips. In general, the smaller the floor radii can be the better. Equal corner radii will easily cost 10 times what unequal corner radii costs.

3. **Material Shape & Size:**

- Design parts so they can be machined within the smallest stock material available. For example, Figure 8 shows a part 3.3" wide by .74" thick. This part can be made from 3-1/2" wide bar stock, but isn't thin enough to be made from 3/4" thick stock because it only leaves .01" of material to allow cleanup of the faces. Material 1" thick would have to be used to ensure complete face cleanup. The thicker raw stock costs 25% more and will take more machining time to remove excess stock. If the part could have been designed to a maximum thickness of .65" or less, then .75" thick raw stock could have been used. If smaller parts will be held by clamping in a vise while machining, .05" of excess material is about the minimum amount of excess material needed to fully cleanup the part.

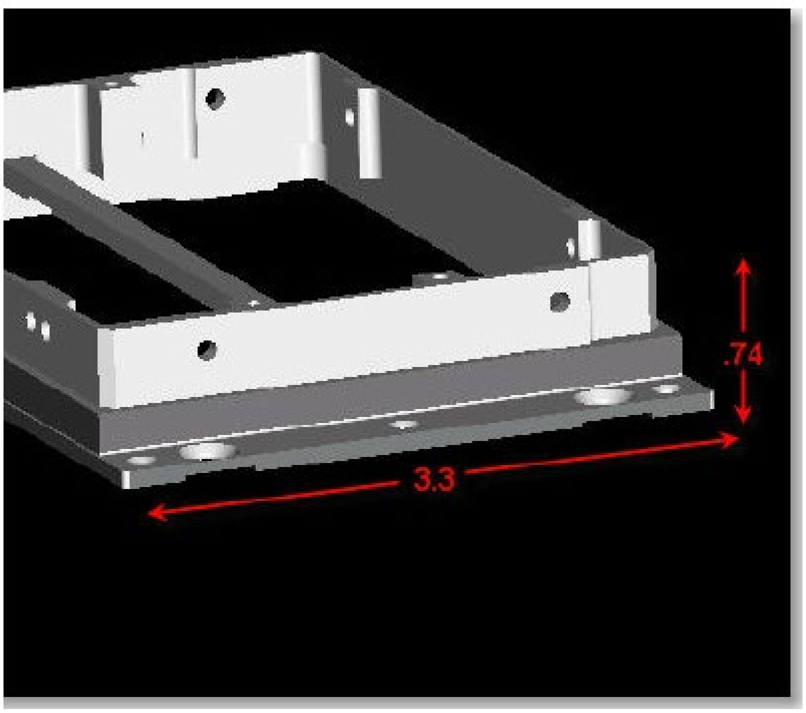
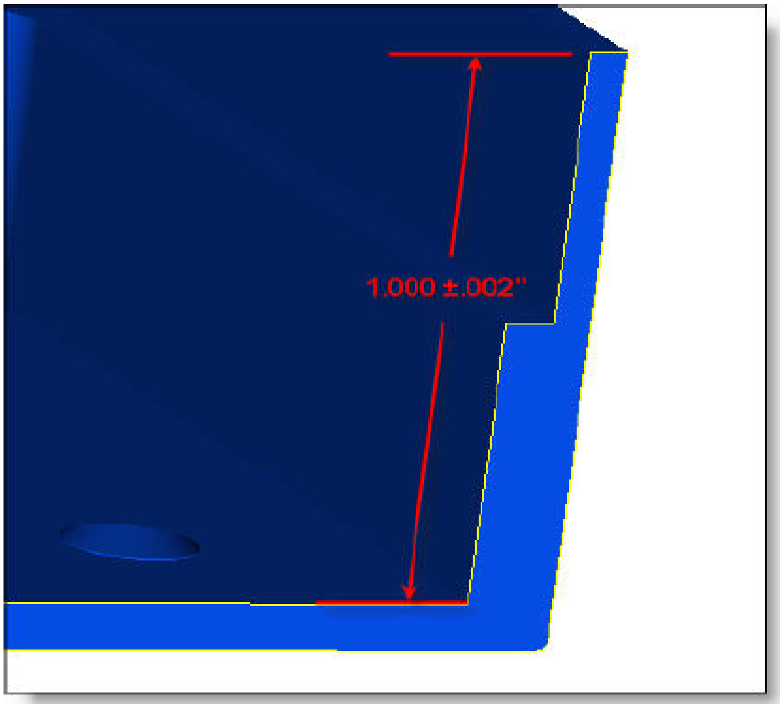


Figure 8 - Source: Pro CNC Inc.

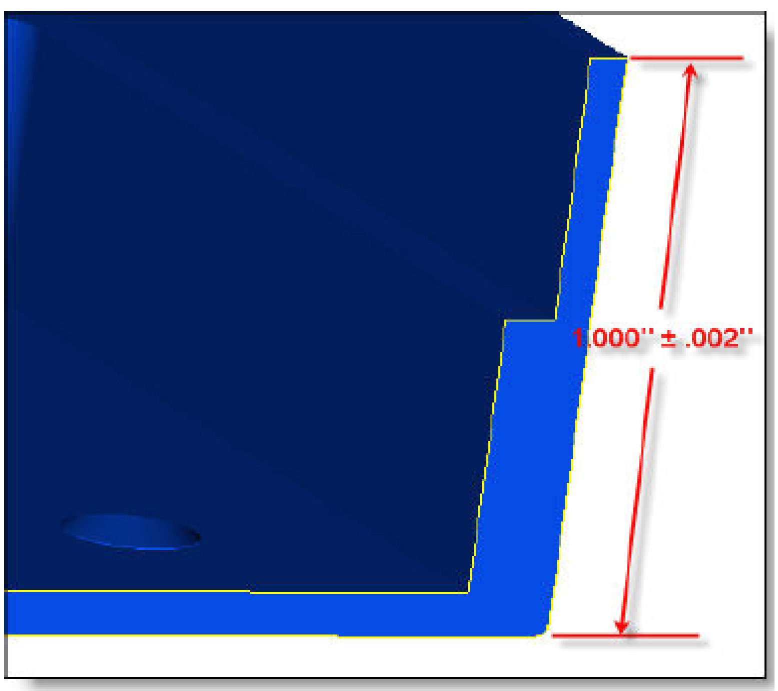
- If all sides of a part need to be finish machined, a general rule of thumb is to have 1" of excess stock material on the length and width and .125" on the thickness.

4. **Tolerances:**

- In general, features that are created by the machine tool capability will be easier to hold to a high tolerance than features which require operator handling and loading into subsequent fixtures. For example, an easy to hold tolerance is the dimension between two steps on the same side of a part (see Figure 9). The same tool will be used on these faces and the positional accuracy of the machine tool will be the primary contributor to variability. On the other hand, if you need to specify a high tolerance to the opposite face of the part (see Figure 10), holding the higher tolerances will be more challenging because the part will have to be removed from the machine, manually flipped upside down, and re-clamped in order to machine the back side.



<u>Figure 9: Faces on the same side are easier to hold to tight tolerances, Source: Pro CNC Inc.</u>



<u>Figure 10: Faces on opposite side are more difficult to hold to tight tolerances, Source: Pro CNC Inc.</u>

- Avoid relying on the sheet block tolerances, such as .xx = .01”, .xxx = .005", .xxxx = .001", to help communicate needs. Sticking with these general tolerances is easy to specify, but can drive fabrication costs higher. For instance, if a .005” tolerance is too loose, .001” isn’t the only other option. Specify something in the middle like .003” or even .0035”. An intermediate tolerance may be easier to hold and subsequently less expensive. When smaller tolerances are specified, a small difference can make it easier to hit the tolerance. A tolerance of .0015” is 50% more tolerance to work with than .001”. The extra .0005” might make the difference in hitting the tolerance requirements on the first try or reworking the part to get it in tolerance, thus driving the cost up.
- It is becoming common to apply a global profile tolerance to an entire part. While easy to specify, this may require significant inspection costs to prove. Even if a part does fall inside a profile tolerance, the manufacturer would need proof of that being the case. This would require a significant number of hand measurements or a full CMM report, either of which is costly. While profile tolerances have a real purpose, be sure you are using them effectively.

5. <u>Threaded Holes:</u>

- The depths of threaded holes are hard to control and costs can increase quickly for threaded holes with tight depth tolerances. The most common mistake is to leave a threaded depth dimension with 3 decimal places so it defaults to the title block tolerance (typically +/- .005”). The reason this is a problem is that taps are not very consistent between their tip and the first full thread. A “bottoming” tap typically has between 1.5 to 2.5 leading threads before the thread profile is complete. Taps can be measured to try to approximate the amount of lead for any specific tap, but it is much easier to specify a looser depth tolerance.
- Avoid #6-32 threads. The ratio of major to minor diameter is greater than other thread sizes. This makes the tap more susceptible to breaking than other thread sizes.
- Don’t specify threaded holes that are deeper than you need. They become expensive as taps are more susceptible to breaking the deeper you go. If the thread depth is deeper than the fastener will thread, money will be wasted on the extra thread depth.
- Figure 11 shows an example of a hole where the minimum distance from the shoulder of the drill to the first full thread is called out. This is a ¼”-20 UNC threaded hole, which would require a minimum of .125” (.05” x 2.5 threads). Any greater than that will not save money and any tighter than that will take more adjusting and cost more.
- Also in Figure 11, the note specifying the tapped hole calls out the most cost effective way to notate this thread on a drawing. The depth is called out as .88 min rather than .875. We also see that there is no pre-drill size specified. Cut and roll taps use different sized predrilled holes so the predrill size should not be specified unless there is a very specific reason for doing so. The countersink diameter specification in this example is also just two decimal places which is much less expensive than using three decimal places. To give an example of how costs can be affected by these tiny details, if this tapped hole callout specified a predrill size, a .875” depth, a Ø .270” countersink and had less than .125" clearance to the drill shoulder, this hole could easily cost 3X - 5X what it would as represented in the tapped hole callout in Figure 11.

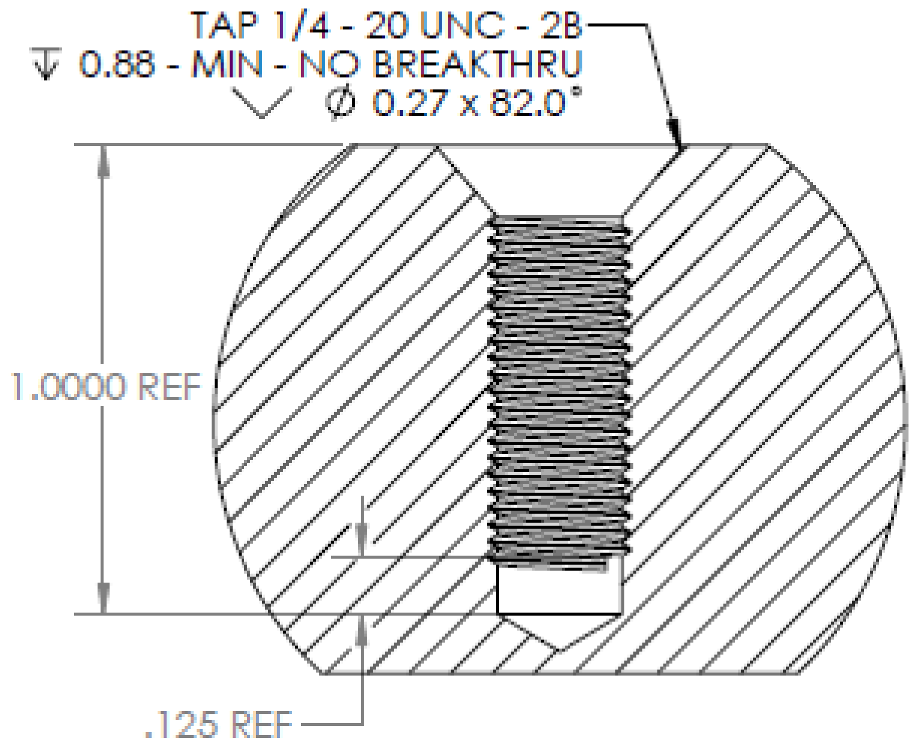


Figure 11: How much clearance to leave and the most cost effective way to specify a threaded hole dimensions.

There is a propensity to install threaded inserts (Helicoils, Keenserts, etc.) in threaded holes machined into specific materials. These are necessary when the material is plastic, but they may also be necessary for some metal applications as well. Threaded inserts are often called out for aluminum parts due to the possibility of the fastener "galling" and damaging the threads. This is more likely to happen if the fastener is installed and removed numerous times over the life of the part. However, if the fastener is going to be installed and left in place or only removed a few times, then a threaded insert is most likely not necessary. Also threaded inserts can increase the strength of the threaded hole. This may be unnecessary if you can engage your fastener at least two times its diameter in threads, then even in aluminum there is a good chance the threads will be stronger than the fastener itself. Depending upon the application, it may be cheaper to occasionally repair threads than to specify all with inserts.

- Estimates of additional cost for installing Helicoils are approximately an extra $1 to $3 per insert. Some parts may have numerous threaded holes in them with Helicoils installed, a B61 aluminum center case is a good example. Each center case has approximately 60 Helicoils installed in them. PF Division recently machined five (5) of these for Sandia National Lab (SNL), so a total of 300 Helicoils had to be installed. At a cost of $3 per Helicoil, the total cost would be $900. If the manufacturer sells these in set quantities, say of 500, then the cost goes to $1,500. Then there is the cost of having a machinist install the Helicoils. For these five center cases it could easily take one machinist a full day to install all the Helicoils. This represents about another $1,500 to the customer. So for this example, the cost of just the Helicoils could easily add $2,400to $3,000 to the total cost. In general, if inserts are necessary, Helicoils are typically cheaper to buy and install.

6. **Total Surface Profile:**

- The current trend is to adopt minimally dimensioned drawings and rely on the CAD model to control feature shape and location. This can lead to misuse of the total surface profile callout. Typically, only a few surfaces on a part are critical to the function of the part. The remainder of the surfaces are much less important. If a minimally dimensioned drawing is being made to manufacture the part, then you would want the less significant surfaces to be controlled by the model. If you dimension the few items you care about on the minimally dimensioned drawing and then specify a callout such as the one shown in Figure 12, you will have increased the inspection requirement by an order of magnitude.

> Undimensioned features are to be controlled by the model within a total surface profile tolerance of .005"

Figure 12: Bad Use of a Global Total Surface Profile Callout, Source: Pro CNC Inc.

By specifying all undimensioned features you are specifying every radius, every chamfer, every hole diameter - every surface on the entire part. Ensuring that all those features are within 0.005" is very expensive to manufacture and measure. It is reasonable to think that with modern CNC machines and CAM programs this requirement would be easy to meet, but it is not. While it is true that the part will be programmed from the CAD model and modern CNC machines are very accurate, there are always obstacles to perfection. The key to solving this problem is to not use a numerical value with your callout. An example is shown in Figure 13. It is the numerical value - no matter what the value is - that is the cost driver. Even if you specified that all undimensioned features only needed to be within 0.05", it still needs to be proven. The reality is that the part will be programmed to the CAD model, and that the part will be very close to the modeled and programmed size; probably well within 0.005" in many cases. If that assurance isn't good enough for any given feature, then you should specify a specific tolerance for that feature.

> Undimensioned features are to be controlled by the model.

- **Figure 13: Good Use of How to Control Undimensioned Features, Source: Pro CNC Inc.**
- When you truly do need to control the profile of a given feature, then make sure you use a profile callout such as:

⌓	.01	A	B

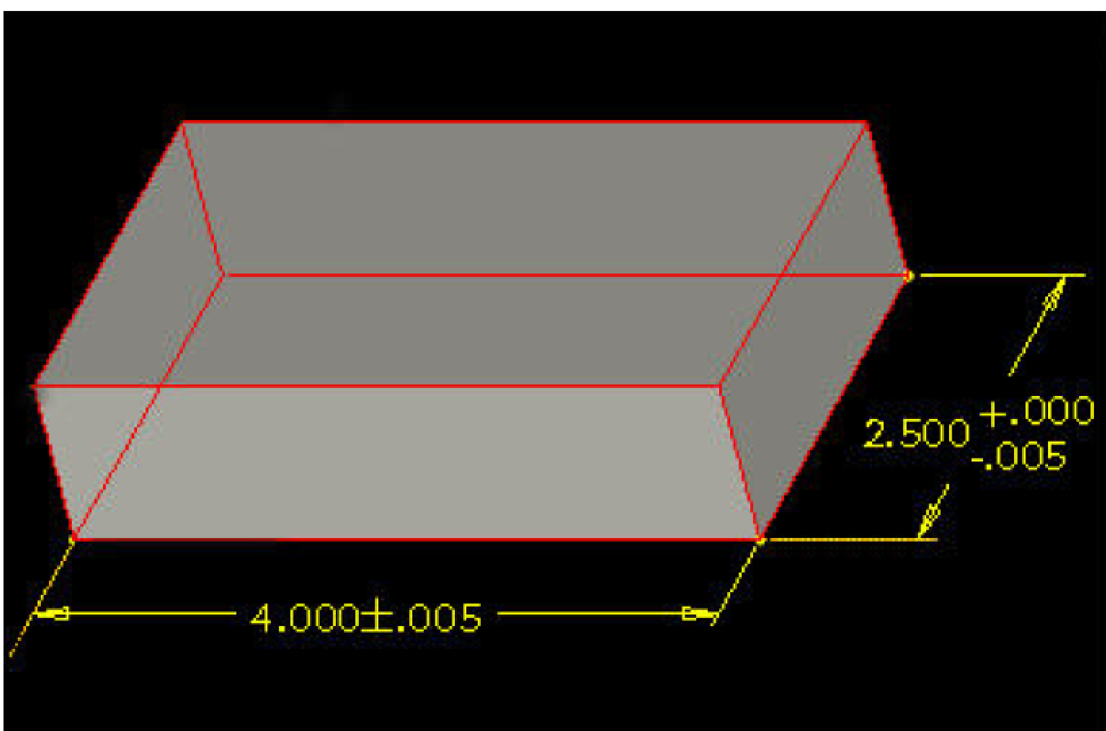
This type of GD&T not only controls the profile of the feature, but with the addition of the datums it also controls the location of the feature to the same tolerance. The addition of the datums can be appropriate if needed, but can also drive cost. Particularly with large and flexible parts, you may consider adding a note about checking the profile in a restrained condition. This may make the inspection significantly easier than in a free state. An alternate method for controlling the profile shape of a feature but allowing more leeway on location is with a compound profile callout such as:

⌓	.01	A	B
⌓	.003		

This will allow .010" locational tolerance to the datums but only .003” on the shape of the feature.

7. **<u>Bilateral vs Unilateral Tolerances:</u>**

- Bilateral tolerancing (also known as symmetric tolerancing) is a method of tolerancing a dimension using equal plus and minus deviations from the nominal dimension. Unilateral tolerances (also known as asymmetric tolerances) on the other hand specify a deviation in only one direction, either plus or minus, from the specified nominal dimension. Unilateral tolerances may also take the form of plus and plus-plus, minus and minus-minus, or plus or minus some amount and plus or minus a lesser amount. Nearly all CNC machines these days are programmed from a 3D CAD model using CAM software. The programmer chooses edges or surfaces on the model to drive toolpaths. Figure 14 shows a basic rectangle with one side having a bilateral tolerance and the other having a unilateral tolerance.



<u>Figure 14 – Examples of Bilateral and Unilateral Tolerancing, Source: Pro CNC Inc.</u>

This is a pretty common example. In this case, the easiest way to program and manufacture this part is with a profile cut from the top of the part which establishes both the 4.000" and 2.500" dimensions at the same time with the same cutter. The CNC program by default will drive the tool at the nominal size of the CAD model. It will create a rectangle that is 4.000" x 2.500". But the sides with a 2.500" dimension will be practically out of tolerance as programmed since we cannot be on the plus side of nominal at all. And with the typical small amount of cutter deflection outward, the 2.500" dimension will likely be out of tolerance. This makes the programming of the part take much longer as the programmer will have to compensate for this with manual adjustments to the program, adjustments to CAM parameters or a separate program with tool paths for each side. The set-up of the part on the CNC machine may also take longer if the machinist needs to apply cutter compensation in order to bring the part closer to the middle of the tolerance zone. In an extreme example, if you had a tolerance of + 0.005"/- 0.000" on the 4.000" dimension, it would be virtually impossible for either dimension to be in spec without major tweaking to the program. This brings up the other point about unilateral tolerances. Most CNC shops want to run the dimensions in the safest range to minimize the chance of scrap. So in the case of the part in Fig. 14, they would likely try to run the 2.500" dimension at 2.4975" which is exactly in the middle of the + 0.000"/ - 0.005" range. This somewhat defeats the purpose of specifying a unilateral tolerance in the first place. It would be easier for everyone involved if the engineer put a symmetric bilateral tolerance at 2.500" or 2.497" because more than likely it is what will be delivered anyway.

- The issue of unilateral tolerances is not as much of an issue with holes (see Figure 15). Very often it is more critical to be able to hold odd tolerances on holes to achieve proper slip, or press fits, etc. Because most holes are machined with drills or reamers the same problem with programming does not exist. The size of the hole will be established with the tool, not the program. So if you need a .2500" hole for a press fit for a 1/4" pin then the shop will select the exact size reamer needed. In the case of an odd sized hole that doesn't match a common drill size or reamer size, the problem may exist to a small degree. A hole such as that can be programmed with a circular interpolation using a smaller end mill to create a larger hole. Since that is more of a "stand alone" type of feature, the parameters can be more easily tweaked to achieve the desired tolerance without affecting other features on the part.

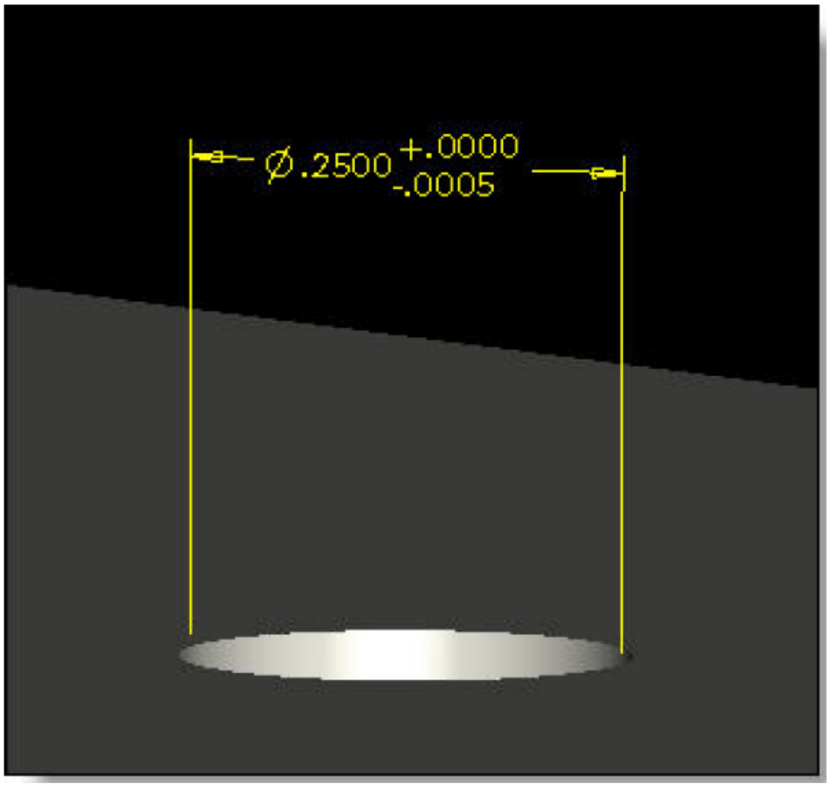


Figure 15 – Unilateral Tolerancing of Holes is not a Problem, Source: Pro CNC Inc.

8. Countersinks and Chamfers:

- In many cases, the diameter of a countersink is not very critical and can often be dimensioned at a lower tolerance than the standard tolerances for the part. Unfortunately, we see all too often that the diameter tolerance is left as a 3 decimal place dimension which would assign a higher tolerance than is needed - generally ±.005". See Figure 16.

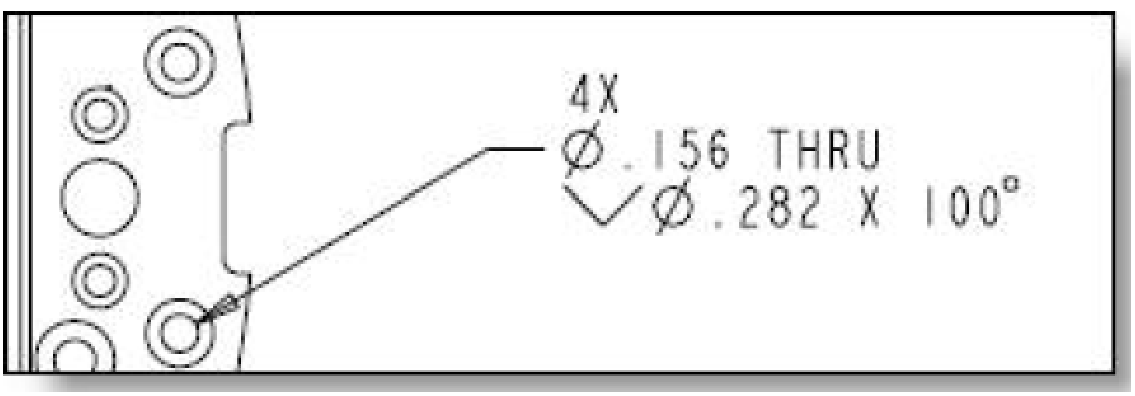


Figure 16 – Expensive Countersink Callout, Source: Pro CNC Inc.

Normally that much tolerance is no problem for CNC machining. So why would this drive additional cost vs. a lower tolerance feature? There are a few reasons why this is a special case. Generally a countersink feature is specified as the diameter of the outer edge and the included angle. This feature is machined with a countersink tool and the Z-depth of the tool will determine how large the countersink is. With a 90° included angle on the tip of the tool, for every .001" change in Z-depth the diameter will change by .002". See Figure 17.

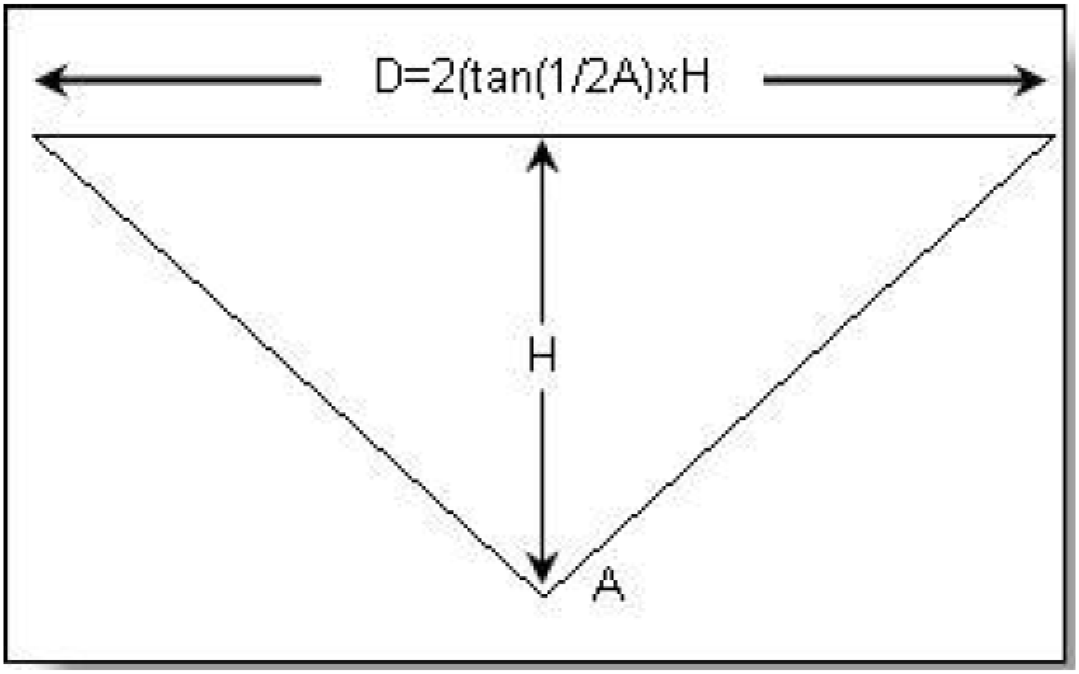


Figure 17 – Diameter versus Depth Relationship for Countersinks, Source: Pro CNC Inc.

With a 100° or 120° tool the condition is further exacerbated - .001" of movement with a 120° tool equates into .0034" change in diameter. So to hold ±.005" on diameter, the Z-depth of the tool needs to be held to ±.0015" which is not easy to do. Even high quality countersink tools are notoriously unreliable on the dimension from their tip to X diameter along their cutting edge. This makes dialing in the tool more challenging the first time and after any tool replacement. Any additional time adds costs to the part. The higher tolerance that you need to hold the diameter to, the longer it will take to dial it in, and the more closely it will need to be watched while running the parts.

- If the countersink is being used for a flathead screw, it should be possible to make the tolerance looser by removing a decimal place as shown in Figure 18.

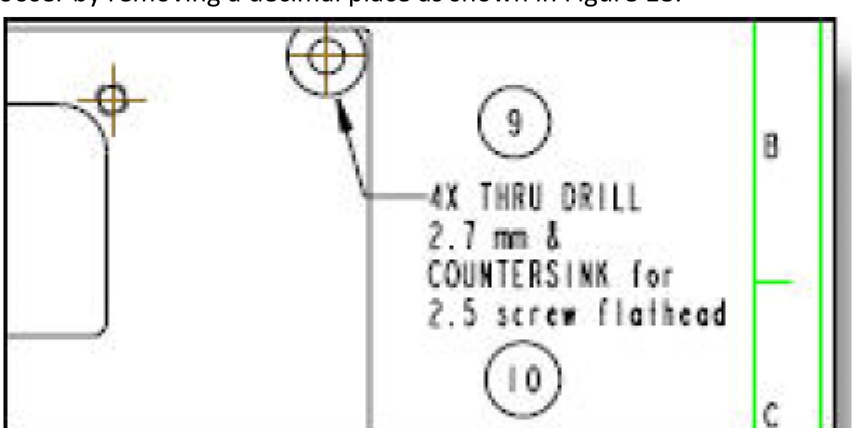


Figure 18 – Callout for Less Expensive Countersink, Source: Pro CNC Inc.

In this example, the countersink tolerance with one decimal place is .5mm or approximately ±.020": a very generous and inexpensive tolerance. If you need to ensure the fastener head is not protruding above the surface, then make the countersink a little deeper. To really ensure it is never protruding above surface, put that exact note on the drawing and specify the part number of the fastener that will be used. Having a "GO" gage is always a positive and quick way to check a feature, particularly if the same feature is machine on a large number of components.

- For machined chamfers that are not on holes, consider the use of the feature and tolerance it accordingly. Again, many engineers forget to loosen the tolerance and the shop is stuck dealing with a more challenging feature and over processing the manufacturing; ultimately, the customer is stuck paying for it.

9. **<u>Designing and Machining Plastic Parts:</u>**

- Below is a list of common plastics that PF Division shops machine:
  - **ABS** comes in both natural (off white) and black and with various levels of glass fill. It is a relatively low-cost plastic that is easy to machine. It holds tolerances reasonably easily and sands and paints well. It has great impact strength and abrasion resistance. Be aware it is also hygroscopic which means it will absorb moisture from the air affecting dimensional stability. The tensile strength is approximately 6 KSI and it is generally available in round bar up to 4" diameter and plate up to 3" thick.
  - **Acetal** or (widely known by the brand name Delrin) is one of the best plastics to use for machined parts. It is a medium-cost plastic with good dimensional stability and excellent machinability. It has very low water absorption which improves dimensional stability. It is available in white, black, various levels of glass fill, or as Delrin AF which has Teflon fibers for increased wear characteristics. It has up to 10 KSI tensile strength and is generally available in round bar up to 6" diameter and plate up to 4" thick.
  - **Acrylic** is also known as PMMA. It is a low-cost plastic that has decent machining characteristics. With the right cutter geometry, very fine finishes can be achieved. It is a relatively hard and rigid plastic which makes it susceptible to chipping; avoid designing thin sections and sharp edges. Model radii and chamfers on outside edges to help reduce the chance of chipping. The main reason to design with acrylic is its excellent light transmission and optical properties. It has good impact strength but not as good as polycarbonate. It has better dimensional stability than many of the softer plastics although it is still susceptible to changing size with temperature fluctuations. It is also slightly hygroscopic but much less than most. Acrylic can be purchased in a MIL-P-5425 grade which is preshrunk to improve its dimensional stability. If acrylic is to be painted after machining, then an additional annealing step is required to ensure it doesn't shrink further when the paint is cured. If threaded inserts are to be installed, it is advisable to rough-machine the material, install the inserts, anneal and then finish machine to reduce the chance of cracking induced by stress of the inserts being installed. Acrylic responds quite well to vapor polishing or flame polishing in applications where machining marks cannot be tolerated. It has about 9 KSI tensile strength and is available in round bar up to 6" diameter and plate up to 2" thick.
  - Nylon has a lot of great properties but comes with several disadvantages for machining. It is a low- (Nylon 6) to medium- (Nylon 6/6) cost plastic. It is pretty strong with tensile strength of about 11-12 KSI, but it is softer than acetal and much more hygroscopic. It tends to warp easily and it seems to move around when you machine it. It is terrible to

deburr as it is very stringy and leaves behind fuzz unless cutters are razor sharp. It does have great toughness, wear, and abrasion resistance which is probably why it is harder to machine. There is also a grade of Nylon called MD or MDS. This grade has molybdenum disulfide in it which makes it more wear resistant than regular nylon and improves the machinability.

  - Polycarbonate has superior impact resistance. It is a medium-cost plastic. It comes in clear and black grades as well as myriad filled grades. It machines well, although like acrylic, can also be susceptible to chipping. It is a pretty stable material with very low water absorption and holds higher tolerances well. It has pretty good thermal resistance and resists deformation up to 265 degrees F. It also vapor polishes very well and can give excellent finishes. It has tensile strength of about 10 KSI and is available in round bar up to 6" and plate up to 2" thick.
  - Ultem is a translucent amber color and is a very high performance engineering thermoplastic. It is an expensive material but offers lots of great properties. It can handle very high temperatures: up to 340 degrees F. It is very stable dimensionally and has very low moisture absorption. It is also rigid but this rigidity contributes to the tendency to chip which is one of its drawbacks for machining. It is also slightly more abrasive than some plastics which increases tool costs. Ultem 1000 is the basic unfilled variety, with 2300 being the 30% glass version. It is extremely strong with about 17 KSI tensile strength and is available in round bar up to 4" and plate up to 2" thick.

- All plastics are less stable than metals. They have much higher thermal expansion and are affected by humidity if they are hygroscopic. These factors need to be taken in to account when designing and tolerancing your part. It is not uncommon to have a machine shop machine and verify a part is in tolerance. A few weeks later, when the parts are inspected, the results are different, which may result in an out-of-tolerance condition. Care should be taken to reduce the chance of this happening. The best solution is to change the geometry to be more stable or increase the tolerance to allow natural variations to occur without becoming out of tolerance. Suggestions for improving the dimensional stability include designing thicker sections, adding ribs, allowing large fillets and adding corner radii. In contrast to injection molding recommendations, it isn't important to have the wall thicknesses be uniform and thin. Unless weight is a big factor, thick solid sections will be more stable and less costly to machine.
- In general it is recommended allowing approximately +/- 0.001" of tolerance per inch of part size. There are some lower performance plastics that would need approximately double that much to be consistently easy to process and stay in tolerance. Because of the higher thermal expansion (up to 10x greater than metal) and lower geometric stability of plastics, it can be more costly to achieve higher tolerances. As parts get larger and sections get thinner, it can become even more difficult to achieve tighter tolerances. Sometimes the measurements themselves can cause deflection in the part which leads to erroneous readings. An example would be measuring a large ring with a caliper where the pressure from the caliper will elongate the ring causing it to appear out of tolerance. Non-contact measurement methods can be employed to reduce this problem.
- If you are designing a round part that is a bearing, sleeve, or some type of part that will mate with a metal component, consider the application of an average diameter callout. It is common for round parts with thin wall thicknesses to “ovalize” which may not matter at all to the function of the part but may cause a big hassle for the part inspector. Alternatively, you can specify the type of fit you desire and provide a representative mating part for inspection.

10. **Raw Stock Sizes:**
- Earlier in this paper there was discussion about how costs can be saved by minimizing the size of raw material needed in order to be able to machine an acceptable part from a given design. Below is a list of common materials that PF Division machines and the typical shapes and sizes of raw stock which should be readily available from distributors :
    - **Aluminum** - In the most common grade - 6061-T6 or -T6511 there are a wide variety of sizes available. In rectangular bar shapes, the thickness generally starts at 1/8" and goes in 1/8" increments up to 1.5" thick and then goes in 1/4" increments up to 6". The widths generally start at 1/2" wide and go in 1/4" increments up to 12" wide depending on the thickness. 2024 and 7075 alloys come in fewer sizes, and tend to follow 1/4" or 1/2" increments. Square bars are generally available in 1/4" increments with some smaller increments available under 1.5". The amount of material that is required to clean up a machined face is a consideration when trying to optimize the size of material that will be used. It is advisable to leave 0.1" on the width dimension for square or rectangular bar stock unless you are expecting to leave the stock surfaces and tolerances in your finished part. On thickness, .125" is the minimum amount of extra material needed, primarily for work holding reasons. But the thicker the part, the more extra material is needed. On a part made from 4" or thicker bar, as much as .25" might be needed to hold on to the part. Consult your manufacturer to see what they suggest. In aluminum round bar, a very wide variety of diameters are available. The sizes start at 1/8" diameter and go in 1/16" increments up to about 2", after which the increments are 1/8" up to about 5", and then 1/4" or 1/2" increments after that up to 20". The amount of excess stock that is needed to clean up is much smaller compared with rectangular shapes. But it varies considerably depending on the diameter of the material and the type of material (i.e.; -extruded vs cold finished). As little as .020" can be anticipated to be cleaned up on the outer diameter for smaller diameter materials. So if you were going to design a part to fit into 1.0" diameter bar stock, then try to make it no larger than .975" on the OD. This is a safe rule of thumb. There are cold finished grades of material which come with much tighter mill tolerances on the OD, allowing even closer dimensions to be achieved. Cold finished aluminum in a 1.0" diameter has a +/- .0025" tolerance, making it possible to design a .990" diameter part to be made from cold finished material. Extruded material on the other hand, has a much looser stock tolerance - +/-.012" on a 1.0" diameter, which is why it is prudent to leave at least .020" for clean-up. The larger the material becomes, the looser the material tolerance, and you need to leave more room for clean-up. 6" diameter extruded aluminum for example has a +/- .044" OD mill tolerance. So it would be advisable to leave at least .062" between the finished part diameter and the raw material. There are so many factors at play, the best option is to ask your manufacturer up front in the design phase so you can optimize the size needed.
    - **Aluminum Plate-** 6061-T651 plate starts at .25" thick and with the exception of .3125", it comes in 1/8" increments up to 1.5" thick and then comes in 1/4" increments up to 3.5". After that, it is available in 1/2" increments up to about 8". With plate, similar to round bar, the thicker the size, the looser the mill tolerance will be and the more you should anticipate will need to be cleaned up. The thickness tolerance range is approximately .023" on 1/2" plate and .075" on 3" plate to give a rough idea.

- **Stainless Steel -** There are such a wide variety of stainless steels, all of which seem to come in different sizes. The more common grades such as 303, 304, 316 and 416 come in nearly every size of round bar you can hope for. In rectangular sizes, the choices are more limited as some alloys come primarily in round and square sizes (for parts typically made on CNC turning centers). Tolerances on cold drawn round, square and rectangular bar are generally very good. Hot rolled variants have much lower tolerances and this needs to be accommodated for in your design.
- **Brass -** Being a popular grade for turning, brass is available in very small increments up to about 6" round bar. The selections for square are decent, but your options are very limited for plate and as far as we know, rectangular sizes are not available at all. Mill tolerances are fairly good so not a lot of excess material is needed to ensure your part gets cleaned up from the raw material.
- **Steel -** There are so many varieties and alloys of steel that it would be an enormous undertaking to describe them all. Similar to stainless steel, tolerances on round, square and rectangular bar are generally very good, especially in the cold drawn varieties. So allowance for material tolerance is not a big factor. Recommendations for rectangular sizes of .1" on width and length and .125" on thickness would generally still apply. Recommendations on round parts are similar to those above for aluminum.

References:

- *"Design Guide" (PDF). Pro CNC. Retrieved January 30, 2017*
- **https://parametric-usa.com/how-to-design-low-price-machined-parts-and-why/**
- http://me.gatech.edu/files/capstone/L071ME4182DFA
- **https://les-electroniciens.com/sites/default/files/cours/smta-gdl_smta**
- **gdl_dfm_presentation_2012-07.pdf**